

3 Parts

The *Part* is the fundamental visualization entity in EnSight. Virtually every postprocessing task you perform will involve a Part, thus it is vital to understand how Parts work.

A Part is a collection of nodes and elements that are grouped together and share the same attributes. When you start EnSight, you either read directly or interactively extract Parts from the data files. Parts which come from the original dataset are referred to as model Parts. Other Parts created within EnSight, are referred to as created (or dependent) Parts.

In this chapter you will learn how to create derived Parts and how to modify the attributes of all Part types.

Section 3.1, [Part Overview](#) is *extremely important*. It defines how Parts work together to form other Parts and explains the dependencies which may exist between model Parts and created Parts. Failure to understand the concept of Parts as explained in this section will limit your ability to use EnSight. Please study this section carefully.

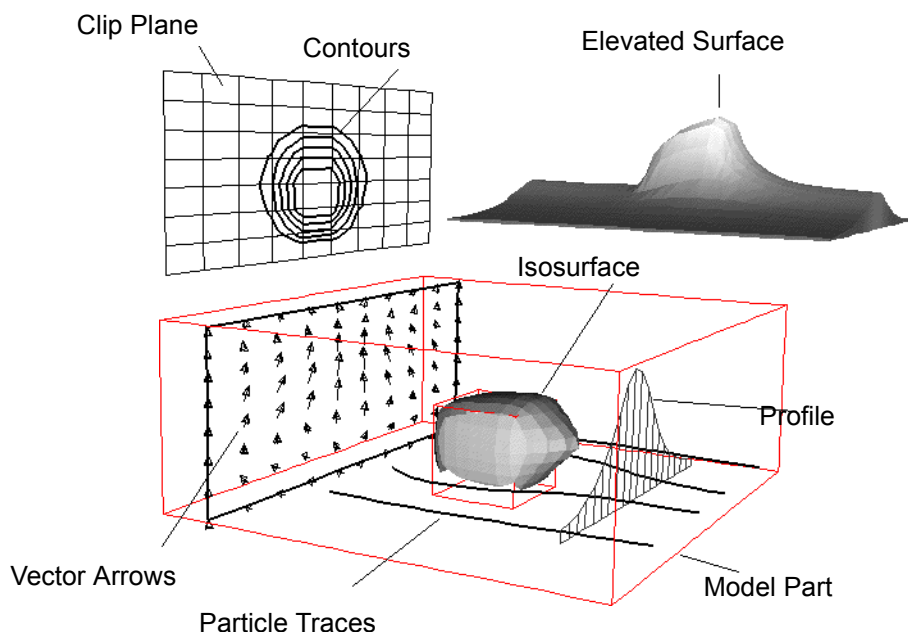


Figure 3-1
Various EnSight Part Types

Included in this Chapter are:

[Section 3.1, Part Overview](#)

[Section 3.2, Part Selection and Identification](#)

[Section 3.3, Part Editing](#)

[Section 3.4, Part Operations](#)

3.1 Part Overview

In EnSight, a Part is simply a collection of nodes and elements which are grouped together, will be manipulated together, and which share the same attributes. This section defines Parts and how they are related. It gives you an overview of the Part types and Part attributes that are available within EnSight.

Parts that are defined or extracted from your dataset are referred to as *model* Parts. Parts that are created within EnSight are referred to as *created* (or *dependent*) Parts. The types of Parts that you create depends on what features within EnSight you choose to utilize. Any created Part is derived from Parts that already exist, which is why the created Parts are sometimes called dependent Parts—they depend on the Parts from which they were created. The Parts that are used to create a dependent Part are referred to as *parent* Parts. Any time that a parent Part changes, its dependent Parts must also change. A parent Part will change when you change its attributes, or modify the current time in the case of transient data.

The Main Parts List contains all Parts that have been read in from your results data or created within EnSight. Displayed are a Part ID Number, a Part symbol, a case number, and a Part description. Table 3-1 lists all of the different types of Parts and their associated symbols. The figure below of the Parts List shows a number of different Part types.

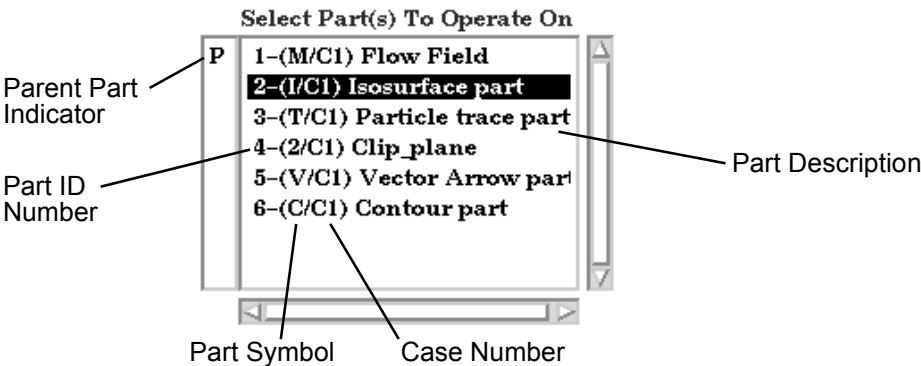


Figure 3-2
Main Parts List

Note that in the illustration above the Isosurface Part is selected and that there is a “P” in the left column next to the Computational mesh (model) Part. This indicates that the Computational mesh Part is the parent Part of the isosurface Part. All parent Parts of a created Part will be so noted if that individual created Part is highlighted in the Main Parts List.

Reassign Parent

Parent Parts of any created Part can be changed by first selecting the created Part in the Feature Detail Editor, then selecting a new parent Part in the Main Parts List, and finally by clicking the Update Parent button in the Feature Detail Editor.

Table 3–1 Part Types, Symbols, and Descriptions

Part Type	Symbol	Description
Clip	(2)	A surface or line resulting from a clip of other Parts using the line, plane, or quadric tools
Contour	(C)	Lines of constant value on 2D elements
Developed Surface	(D)	A planar surface derived by unrolling a surface of revolution (i.e., the unrolling of a cylinder clip Part produced by the cylinder quadric tool)
Elevated Surface	(E)	Surface created by elevating elements by a variable
Isosurface	(I)	Surface of constant value through 3D elements of other Parts
Model Part	(M)	A Part that originated from the dataset
Particle Trace	(T)	Path of a massless Particle through a vector field
Profile	(P)	Plot of a variable along a line (Similar to a 2D elevated surface)
Separation/ Attachment Line	(L)	Line where flow separation or attachment is occurring
Shock Surface/ Region	(K)	Surface or region where shock is occurring
Subset	(S)	Valid node and/or element label range(s) from model Part(s)
Tensor Glyph	(G)	Glyph showing direction of first, second, and third eigenvectors of a tensor field.
Vector Arrow	(V)	Arrows showing direction and magnitude of vector field
Vortex Core	(X)	Line representing center of a flow vortex

Part Creation

Part creation occurs on either the server or the client. Since the data that is available on the client and server are different, it is useful to understand where Parts are created and where the data structures are stored. By understanding this, you will understand why some Parts can be created with certain parent Parts and others cannot. This information can be gained by examining the following table.

Table 3–2 Part Creation and Data Location

Part Type	Where Created	Data on Server	Data on Client
Clip	Server	Yes	Depending on Part attributes
Contour	Client	No	Yes
Developed Surface	Server	Yes	Depending on Part Attributes
Discrete Particle	Not Applicable	Yes	Depending on Part attributes
Elevated Surface	Server	Yes	Depending on Part attributes
Isosurface	Server	Yes	Depending on Part attributes
Model	Not Applicable	Yes	Depending on Part attributes
Particle Trace	Server	No	Yes
Profile	Client	No	Yes

Part Type	Where Created	Data on Server	Data on Client
Separation/ Attachment Line	Server	Yes	Depending on Part attributes
Shock Surface/ Region	Server	Yes	Depending on Part attributes
Subset	Server	Yes	Depending on Part attributes
Tensor Glyph	Client	No	Yes
Vector Arrow	Client. Server if necessary.	Maybe	Yes
Vortex Core	Server	Yes	Depending on Part attributes

(see [Introduction to Part Creation](#))

Part Attributes

Each type of created Part has a unique set of attributes that are used to accomplish its creation, the *Creation Attributes*. Model Parts (symbol: M) and discrete Particle Parts (symbol: D) typically do not have creation attributes because they are not created—they are read or extracted from the dataset. The one exception to this is model parts originating from block structured datasets. These parts contain the I,J,K and step attributes used to create the part.

All Parts have a set of *Display Attributes* that are used in visualizing the Part in the Graphics Window. These can be modified using the Feature Detail Editor or by utilizing the Part Mode Icons (See Section 8.4). The Feature Detail Editor for each Part type will show you attributes grouped together under turndown sections. They deal with such things as color, line width, symmetry operations, etc. Display attributes do not control how the Part is created, only how it appears or how it behaves in the Graphics Window.

Table 3–3 Display Attribute Sections

Section:	Includes controls for...
General Attributes (see Section 3.3 Part Editing)	Visibility in Graphics Window and individual Viewports Symmetry options Susceptibility to Auxiliary Clipping Reference frame Response to changes in time (frozen or active) Coloration (constant or by a palette associated with a variable) Shaded Surface and Hidden Line display Surface shading (flat, Gouraud, smooth) Opacity and Fill density Lighting (diffuse, shininess, and highlight intensity)

Section:	Includes controls for...
Node, Element, and Line Attributes (see Section 3.3, Part Editing)	General Visibility: Node, Line, and Element Label Visibility: Node and Element Node Representation: Node type (dot, cross, or sphere), Node Scale, Node Detail (for spheres), and Node size (constant or variable) Line Representation: Line Width and Line style (solid, dotted, or dot-dash) Element representation on client (full, border, 3D border/2D full, feature angle, or non visual), Element-size, Shrink-Factor, and Element Angle
Displacement Attributes (see Section 3.3, Part Editing)	Displacement variable Displacement scaling factor

3.2 Part Selection and Identification

In the process of creating a Part you will need to be able to select the parent Part(s) from the Main Parts List. You will also find that it is possible to either read or create so many Parts within EnSight that you become confused as to the identity of each Part. This section describes Part selection and identification

Selecting Parts

Items in all Parts Lists are selected using standard Motif/Win32 methods:

To:	Do This:	Details
Select an item	Select (or single-click)	Place the mouse pointer over the item and click the left mouse button. The item is highlighted to reflect the “selected” state.
Extend a contiguous selection	Select-drag	Place the mouse pointer over the first item. Click and hold the left mouse button as you drag over the remaining items to be selected. Only contiguous items may be selected in this fashion.
Extend a (possibly long) contiguous selection	Shift-click	Select the first item. Place the mouse pointer over the last item in the list to be selected. Press the shift key and click the left mouse button. This action will extend a selection to include all those items sequentially listed between the first selection and this one.
Extend a non-contiguous selection	Control-click	Place the mouse pointer over the item. Press the control key and click the left mouse button. This action will extend a selection by adding the new item, but not those in-between any previously selected items.
De-select an item	Control-click	Place the mouse pointer over the selected item. Press the control key and click the left mouse button. This action will de-select the item.
Open the Quick Interaction Area for a Part	Double-click	Place the mouse pointer over the item and click the left mouse button twice in rapid succession.

(see [How To Select Parts](#))

Identifying Parts

There are two quick ways to identify one or more Parts that have been selected in the Main Parts List. You can identify them in the Graphics Window by toggling visibility on/off while in Part Mode or you can select View > Show Selected Parts... from the Main Menu to show only the selected Part(s) in the pop-up Selected Part(s) Window.

3.3 Part Editing

In EnSight, new Parts can be created and edited in the Quick Interaction Area Editor or in the Feature Detail Editor specific to each type of Part. This process is described in Sections 7.2 to 7.9. For editing, the Quick Interaction Area provides access to the most common attributes; the Feature Detail Editor allow modification of all attributes.

Whereas each individual change made in the Quick interaction Area Editor is applied to the Part immediately, the Feature Detail Editor allows you to make a number of changes to various attributes and then apply them all at one time. This is done by toggling off View > Immediate Modification in the Feature Detail Editor. The default behavior is to immediately apply a change when you press Return.

The Feature Detail Editor for Parts is opened from the Main Menu (or by double clicking on a Part creation Icon in the Main GUI Feature Icon Bar).

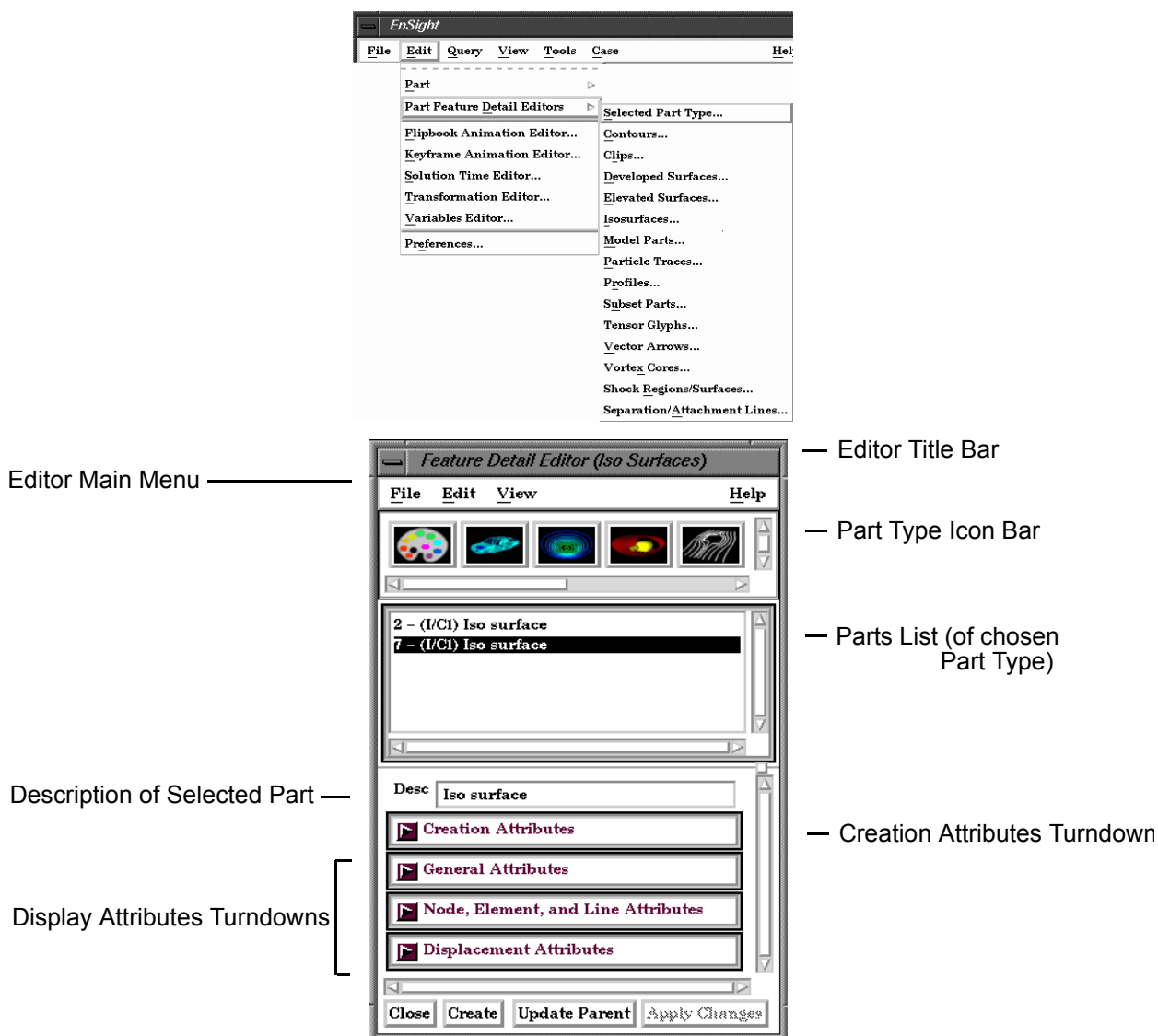


Figure 3-3
Feature Detail Editor (Isosurfaces)

Feature Detail Editor Main Menu

File Not applicable when Feature Detail Editor is used for Parts - only applicable for Variables.

Edit Opens a pull down menu.

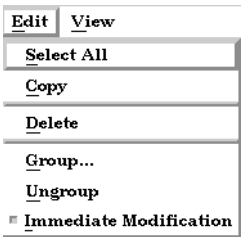


Figure 3-4
Feature Detail Editor Edit pull-down menu

Select All Selects all Parts in Feature Detail Editor Parts List. (see Section 3.4, Part Operations)

Copy Makes a copy of all selected Parts. (see Section 3.4, Part Operations), also (see How To Copy a Part)

Delete Deletes selected Parts. (see Section 3.4, Part Operations), also (see How To Delete a Part)

Group... Groups the selected parts into a new part and removes the original parts from the list.

Ungroup Extracts the original parts out of a group and removes the group part.

Immediate Modification Toggle Toggles on/off the immediate modification of Parts when individual changes are made to Attributes within the Feature Detail Editor. Default is on. By toggling off, you can make several changes within the Feature Detail Editor and then apply them all at one time by clicking the Apply Changes button.

View Opens a pull-down menu.



Figure 3-5
Feature Detail Editor View pull-down menu

Show Selected Part(s) Toggle Opens the Selected Part(s) Window in which only Parts selected in the Feature Detail Editor's Parts List are visible.

Part Type Icon Bar

The Feature Detail Editor is initially opened from EnSight's Main Menu (or by double clicking a Part creation icon in the Feature Icon Bar) and the Feature Detail Editor's Parts List contains all Parts of the type named in the Editor's Title Bar. The type of Parts in the Feature Detail Editor's Parts List may be changed by clicking on the appropriate icon in the Feature Detail Editor's Part Type Icon Bar. The figure below shows the choices available.

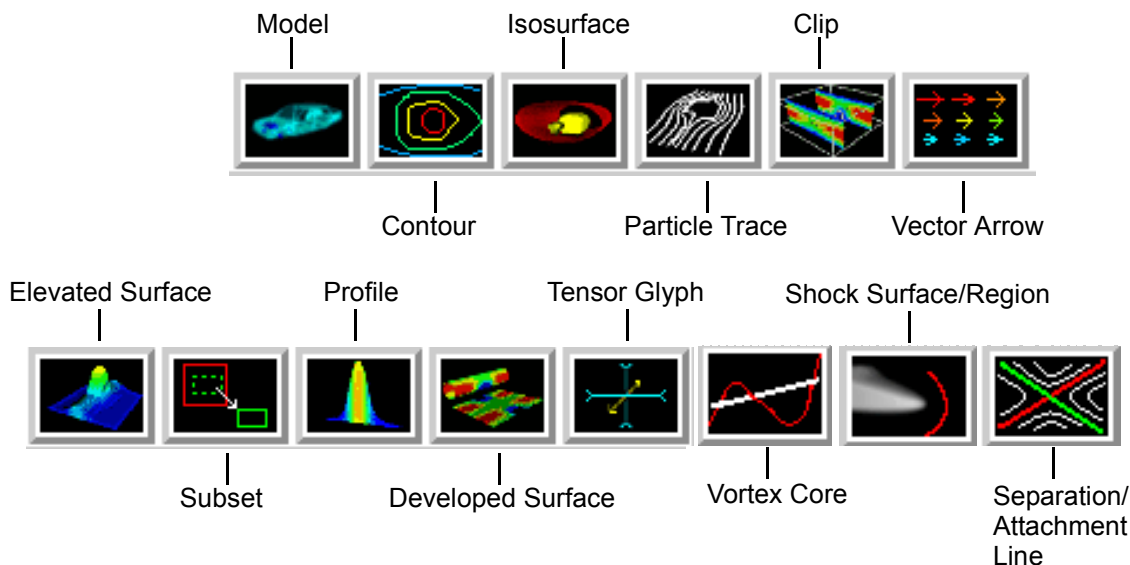


Figure 3-6
Feature Detail Editor Part Type Selection Icons

There is a Color Icon in the Feature Detail Editor's Part Type Icon Bar which, if clicked will open the Feature Detail Editor for Variables.

See Section 4.1 Variable Selection and Activation for further discussion.

Creation Attributes

Creation Attributes are "specific" attributes used to create (or modify) model and created Parts.

Model Parts

Creation Attributes for updating the I,J,K node range attributes of the selected block structured Model Parts with proper updating of all dependent parts and variables. The Creation Attributes area is inactive for unstructured Model Parts.

Access: Main Menu > Edit > Part Feature Detail Editors > Model Parts

The image shows a dialog box titled "Creation Attributes" with a checked checkbox. Below the title is a section labeled "Using Node Ranges:" followed by a table with five columns: "From", "To", "Step", "Min", and "Max". There are three rows of input fields corresponding to the I, J, and K directions.

	From	To	Step	Min	Max
I					
J					
K					

Figure 3-7
Feature Detail Editor (Model) Creation Attributes Area

Using Node Ranges

IJK From These fields specify the desired minimum interval value in the respective IJK

component direction of the Model Part.

IJK To These fields specify the desired maximum interval value in the respective IJK component direction of the Model Part.

IJK Step These fields specify the desired interval stride value in the respective IJK component direction of the Model part.

IJK Min These fields verify the minimum interval limit in the respective IJK component direction of the Model part.

IJK Max These fields verify the maximum interval limit in the respective IJK component direction of the Model part.

(see [How To Create IJK Clips](#))

Created Parts See the appropriate Section in Chapter 7 for a description of the Creation Attributes section.

- (see Section 7.2, Contour Create/Update)
- (see Section 7.3, Isosurface Create/Update)
- (see Section 7.4, Particle Trace Create/Update)
- (see Section 7.5, Clip Create/Update)
- (see Section 7.6, Vector Arrow Create/Update)
- (see Section 7.7, Elevated Surface Create/Update)
- (see Section 7.8, Profile Create/Update)
- (see Section 7.9, Developed Surface Create/Update)
- (see Section 7.16, Subset Parts Create/Update)
- (see Section 7.17, Tensor Glyph Parts Create/Update)
- (see Section 7.18, Vortex Core Create/Update)
- (see Section 7.19, Shock Surface/Region Create/Update)
- (see Section 7.20, Separation/Attachment Lines Create/Update)

General Attributes

General Attributes are “general” in that: (a) all Parts have them, and (b) they can’t be neatly categorized into any other attribute type. Like all Part attributes, they are set individually for each Part.

Access: Main Menu > Edit > Part Feature Detail Editors > General Attributes



Figure 3-8
Feature Detail Editor General Attributes Area

Visible Toggle

Toggles-on/off whether Part is visible on a global basis (in the Graphics Window or in all

viewports). (Performs the same function as the Visibility Toggle in the Parts Mode Icon Bar). Default is ON.

Visible In Viewports

This small window allows you to control the visibility of the selected Part(s) on a per Viewport basis. Each visible viewport is shown. A green outline around a Viewport indicates that the selected Part(s) will be visible in this Viewport, while a red outline indicates that the selected Part(s) will not be visible. Change the visibility (red to green, green to red) by selecting a viewport with the mouse.

Fast Display Rep.

This button opens a pop-up menu button for the selection of the fast display representation used to display a part on the client. This attribute helps the display of complex data sets. The part's fast display representation displays according to whether the Fast Display Mode (located in the View Menu or on the desktop) is on or off and on the state of the Static Fast Display button located under Edit > Preferences..., Performance. For instance, when the Fast Display Mode is Off (default) the part displays according to its specified Element Representation. When on, the parts are displayed by the fast display representation. The fast display representation will only be used while performing transformations, unless the Static Fast Display option has been selected. The part detail representations are:

Box a bounding (Cartesian extent) box of all part elements (default).
Off display according to specified Element Representation.
Points point cloud representation of the part.

(see [How To Set Global Viewing](#))

Symmetry

This button opens a pull-down menu which allows you to toggle-on/off the display of a mirror image of Parts (which are selected in the Feature Detail Editor's Parts List) in each of the seven other quadrants of the Part's local frame. It also allows you to turn on or off the original (non-symmetric) part representation. It performs the same function as the Symmetry Pull-down Icon in the Part Mode Icon Bar. You can mirror the Part to more than one quadrant. If the Part occupies more than one quadrant, each portion of the Part mirrors independently. Symmetry enables you to reduce the size of your analysis problem while still visualizing the "whole thing." Symmetry affects only the displayed image, not the data, so you cannot query the image or use the image as a parent Part. However, you can create the same effect by creating dependent Parts with the same symmetry attributes as the parent Part. Symmetry works as if the local frame is Rectangular, even if it is cylindrical or spherical. The images are displayed with the same attributes as the Part. For each toggle, the Part is displayed as follows. The default for all toggle buttons is OFF, except for the original representation - which is ON.

Mirror X	quadrant on the other side of the YZ plane.
Mirror Y	quadrant on the other side of the XZ plane.
Mirror Z	quadrant on the other side of the XY plane.
Mirror XY	diagonally opposite quadrant on the same side of the XY plane.
Mirror XZ	diagonally opposite quadrant on the same side of the XZ plane.
Mirror YZ	diagonally opposite quadrant on the same side of the YZ plane.
Mirror XYZ	quadrant diagonally opposite through the origin.
Original	the original part location.

(see [How To Set Symmetry](#))

Aux Clip Toggle

Toggles-on/off whether Part(s) selected in the Part List of the Feature Detail Editor will be affected by the Auxiliary Clipping Plane feature, which enables you to make invisible that portion of each Part on the negative side of the current position of the Plane Tool. Performs the same function as the Part Mode: Auxiliary Clipping Toggle Icon. A Part with its Aux Clip attribute toggled-off will not be cut away. Default is

ON. (see Auxiliary Clipping in [Section 6.4, View Menu Functions](#)).

<i>Active Toggle</i>	Toggles-on/off whether or not display of the Part automatically updates as the solution time changes. When visualizing transient data, you may wish to “freeze” a Part in time while other Parts continue to update. For example, you can create two identical vector-arrow Parts, toggle-off Active for one of them, change the time step of the display, and see how the vector arrows change from one time step to the other. Only the EnSight client Part is frozen, the EnSight server Part is kept current. Default is ON.
<i>Ref. Frame</i>	This field specifies which frame the Part is assigned to. Default is the frame of the Part’s parent Part (Frame 0 for original model Parts). Enter a different frame number in the field to change the assignment. Changing a Part’s frame causes the Part to be drawn in the new coordinate frame. Once assigned to a different frame, the Part will transform with that frame. The choice of frame does not affect variable values. The interpolated value of a variable at point 0,0,0 in Frame 0 is the same as at point 0,0,0 in Frame 1, even though the points may appear at different locations in the Main View Window. (see Section 8.6, Frame Mode)
<i>Color By</i>	This button opens a pop-up menu for the selection of the variable color palette by which you wish to color the selected Part(s). Coloring a Part with a palette does not normally affect graphics performance while in line drawing mode, but Shaded Surface mode performance can become considerably slower. If you do not color by a palette (Color By > None), the Part will be displayed according to the color specified in the R, G, B fields. If you want to color Parts by palettes and want Shaded Surface mode, consider using the Static Lighting option (see Static Lighting in Section 6.4, View Menu Functions).
<i>R G B</i>	These fields allow you to specify a solid color for the selected Part(s) (applicable only if Color By is None). Enter a numerical value from 0 to 1 for each component color (Red, Green, and Blue).
<i>Mix...</i>	Opens the Color Selector dialog for the selection of a solid color for the selected Part(s) (applicable only if Color By is None). (see Section 7.1, Color)
<i>Surface</i>	
Shaded Surface Toggle	Toggles on/off surface shading for individual Parts. When global Shaded Surface has been toggled on for the Graphics Window display (from Main Menu > View > Shaded Surface or via the Global Shaded Surface Toggle in the View Mode Icon Bar), individual Parts can be forced to stay in line drawing mode using this toggle. Default is ON. (see Section 6.4, View Menu Functions)
Hidden Line Toggle	Toggles on/off hidden line representation for individual Parts. When global <i>Hidden Line</i> has been toggled on for the Graphics Window display (from Main Menu > View > Hidden Line or via the Global Hidden Line Toggle in the View Mode Icon Bar), individual Parts can be forced not to appear as Hidden Line representation using this toggle. (To have lines hidden behind surfaces, Parts must have surfaces, i.e. 2D elements) Default is ON. (see Section 6.4, View Menu Functions)
Shading	Opens a pop-up menu for selection of appearance of Part surface when Shaded Surface is on. Normally the mode is set to Gouraud, meaning that the color and shading will interpolate across the polygon in a linear scheme. You can also set the shading type to Flat, meaning that each polygon will get one color and shade, or Smooth which means that the surface normals will be averaged to the neighboring elements producing a “smooth” surface appearance. Not valid for all Part types. Options are:

<i>Flat</i>	Color and shading same for entire element
<i>Gouraud</i>	Color and shading varies linearly across element
<i>Smooth</i>	Normals averaged with neighboring elements to simulate smooth surfaces

Opaqueness This field specifies the opaqueness of the selected Part(s). A value of 1.0 indicates that the Part is fully opaque, while a value of 0.0 indicates that it is fully transparent. Setting this attribute to a value other than 1.0 can seriously affect the graphics performance.

Fill Pattern Opens a pop-up menu for selection of a fill pattern which can provide pseudo-transparency for shaded surfaces. Default is Fill 0 which uses no pattern (produces a solid surface), while Fill patterns 1 through 3 produce a EnSight defined fill pattern.

Lighting

Diff This field specifies diffusion (minimum brightness or amount of light that a Part reflects). (Some applications refer to this as *ambient* light.) The Part will reflect no light if value is 0.0. If value is 1.0, no lighting effects will be imposed and the Part will reflect all light and be shown at full color intensity at every point. To change, enter a value from 0 to 1.

Shin This field specifies shininess. You can think of the shininess factor in terms of how smooth the surface is. The larger the shininess factor, the smoother the object. A value of 0 corresponds to a dull finish and a value of 100 corresponds to a highly shiny finish. To change, enter a value from 0 to 100.

H Int This field specifies highlight intensity (the amount of white light contained in the color of the Part which is reflected back to the observer). Highlighting gives the Part a more realistic appearance and reveals the shine of the surface. To change, enter a value from 0 to 1 with larger values representing more white light. Will have no effect if Shin parameter is zero.
(see [How To Set Attributes](#))

Troubleshooting Surface Attributes and Lighting

Problem	Probable Causes	Solutions
Part not in Shaded Surface mode	Global Toggle not on, or if on, Shaded Surface is turned off for the Part in the Feature Detail Editor	Turn on Shaded Surface toggle from View menu of Main Menu or turn and make sure Shaded Surface is turned on for the Part in the Feature Detail Editor.
	Part contains only 1D elements	No Solution
Part appears not to have any lighting.	Diffuse light intensity too high	Lower the Diff value.

Node, Element, and Line Attributes

Each Part’s Node, Element, and Line attributes control the representation of the Part on the client, and how nodes, elements, and lines are displayed.
Access: Main Menu > Edit > Part Feature Detail Editors > Node, Element, and Line Attributes

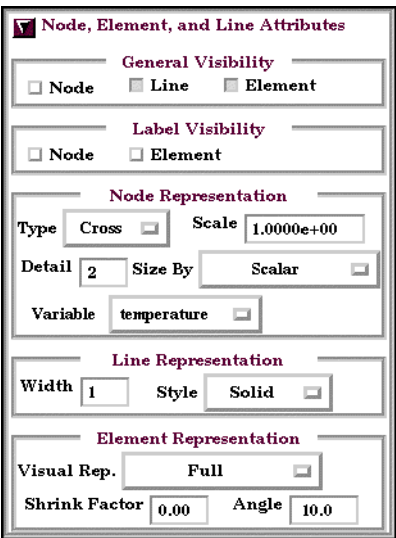


Figure 3-9
Feature Detail Editor Node, Element, and Line Attributes Area

General Visibility

- Node Toggle

Toggles-on/off display of Part’s nodes whenever the Part is visible. Default is OFF.
- Line Toggle

Toggles-on/off display of line (1D) elements in the client-representation whenever the Part is visible. Default is ON.
- Element Toggle

Toggles-on/off display of 2D elements in the client-representation whenever the Part is visible. Note that 3D elements are always represented as 2D elements on the client. Default is ON.

Label Visibility

- Node Toggle

Toggles-on/off display of Part’s node labels (if they exist) whenever the Part is visible. Only model Parts may have node labels. Default is OFF.
- Element Toggle

Toggles-on/off display of Part’s element labels (if they exist) whenever the Part is displayed in Full visual representation. Only model Parts may have element labels, and. Default is OFF.

Node Representation

- Type

Opens a pop-up menu for the selection of symbol to use when displaying the Part’s nodes. Default is Dot. Options are:
Dot to display nodes as one-pixel dots.
Cross to display nodes as three-dimensional crosses whose size you specify.
Sphere to display the nodes as spheres whose size and detail you specify.

Scale	This field is used to specify scaling factor for size of node symbol. Values between 0 and 1 reduce the size, factors greater than one enlarge the size. Not applicable when node-symbol Type is Dot. Default is 1.0.
Detail	This field is used to specify how round to draw the spheres when the node-symbol type is Sphere. Ranges from 2 to 10, with 10 being the most detailed (e.g., roundest spheres). Higher values take longer to draw, slowing performance. Default is 2.
Size By	Opens a pop-up menu for the selection of variable-type to use to size each node-symbol. For options other than Constant, the node-symbol size will vary depending on the value of the selected variable at the node. Not applicable when node-symbol Type is Dot. Default is Constant. Options are: <i>Constant</i> sizes node using the Scale factor value. <i>Scalar</i> sizes node using a scalar variable. <i>Vector Mag</i> sizes node using magnitude of a vector variable. <i>Vector X-Comp</i> sizes node using magnitude of X-component of a vector variable. <i>Vector Y-Comp</i> sizes node using magnitude of Y-component of a vector variable. <i>Vector Z-Comp</i> sizes node using magnitude of Z-component of a vector variable.
Variable	Selection of variable to use to size the nodes. Activated variables of the appropriate Size By type are listed. Not applicable when node-symbol Type is Dot or Size By is Constant.

Line Representation

Width	Specification of width (in pixels) of line elements and edges of 2D elements whenever they are visible. Range is from 1 to 20. Default is 1. Line widths other than 1 are not available on all hardware. This performs the same function as the Part Line Width Pulldown Icon in Part Mode.
Style	Selection of style of line when lines are visible. Default is Solid. Options are: <i>Solid</i> <i>Dotted</i> <i>Dot-Dash</i>

Element Representation

Visual Rep.	Selection of representation of Part's elements on the client. Saves memory and time to download. <i>3D border, 2D full</i> represents the Part's 3D elements in Border representation, the Part's 1 and 2D elements in Full representation. The result is the outside surfaces of the Part are displayed along with all bar elements. <i>Border</i> represents the Part's 3D elements with 2D elements corresponding to unshared element faces, the Part's 2D elements with 1D elements corresponding to the unshared edges, and the Part's 1D elements as 1D elements. The result is the outside faces and edges of the Part's elements. <i>Feature Angle</i> first runs the 3D border, 2D full representation to get a list of 1 and 2D elements. The 1D elements and all non-shared 2D edges will be shown, but only the shared edges above the Angle value will be shown. The result consists of 1D elements visualizing the sharp edges of the Part. <i>Bounding Box</i> represents all Part elements as a bounding box surrounding the Cartesian extent of the elements of the Part. <i>Full</i> represents all faces of the Part's 3D elements, and all the 1 and 2D elements. <i>Non Visual</i> means the Part exists on the server, but is not loaded on the client. Not Loaded Parts may be used as parent Parts, but do not exist on the client.
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Shrink Factor	Specification of scaling factor by which to shrink every element toward its centroid. Enter the fraction to shrink by in range from 0 to 1. Default is 0.0 for no shrinkage.
Angle	Specification of lower limit for not displaying shared edges in Feature Angle Representation. Value is in degrees.

(see [How To Set Attributes](#) and [How To Display Labels](#))

Troubleshooting Node, Element and Line Attributes

Problem	Probable Causes	Solutions
After changing to Feature Angle representation, the Part is not shown.	Angle value is to large	Set Angle to smaller value.

Displacement Attributes

Displacement Attributes specify how to displace the Part nodes based on a vector variable. Each node of the Part is displaced by a distance and direction corresponding to the value of a vector variable at the node. The new coordinate is equal to the old coordinate plus the vector times the specified Factor, or:

$$C_{new} = C_{orig} + \text{Factor} * \text{Vector},$$

where C_{new} is the new coordinate location, C_{orig} is the coordinate location as defined in the data files, Factor is a scale factor, and Vector is the displacement vector.

You can greatly exaggerate the displacement vector by specifying a large Factor value. Though you can use any vector variable for displacements, it certainly makes the most sense to use a variable calculated for this purpose. Note that the variable value represents the *displacement* from the original location, not the *coordinates* of the new location.

Access: Main Menu > Edit > Part Feature Detail Editors > Displacement Attributes

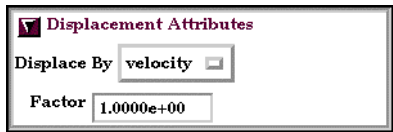


Figure 3-10
Feature Detail Editor Displacement Attributes Area

Displace by Opens a pop-up menu for selection of vector variable to use for displacement (or None for no displacement). Variable must be a vector and be activated.

Factor This field is used to specify a scale factor for the displacement vector. New coordinates are calculated as: $C_{new} = C_{orig} + \text{Factor} * \text{Vector}$, where C_{new} is the new coordinate location, C_{orig} is the original coordinate location as defined in the data file, Factor is a scale factor, and Vector is the displacement vector. Note that a value of 1.0 will give you “true” displacements.
(see [How To Display Displacements](#))

Troubleshooting Displacement Attributes

Problem	Probable Causes	Solutions
Displacement not visible	Displace By attribute set to None for Part that is not displacing. Factor value too small.	Set the Displace By attribute Specify a larger Factor.
<i>Create</i>	Clicking this button creates a new Part using attributes currently selected/specified in the Feature Detail Editor. This performs the same function as the Create button in the Quick Interaction Area Editor for each type of created Part. Clicking Create updates the Graphics Window and adds the new Part to the Main Parts List and to the Parts List in the Feature Detail Editor for this type of Part. Not applicable for model Parts or discrete Particles. (see Introduction to Part Creation)	
<i>Update Parent</i>	Clicking this button assigns the Part which is currently selected in the Main Parts List as the new parent Part of the created Part(s) which is(are) currently selected in the Feature Detail Editor's Parts List.	
<i>Apply Changes</i>	Clicking this button applies all changes that have been made within the Feature Detail Editor all at once if Immediate Modification has been toggled off above in the Feature Detail Editor's Edit pull-down menu. If Creation attributes have been changed, the Part will be regenerated.	

3.4 Part Operations

This section will describe the Part operations accessible through “Edit > Part” in the Main Menu and “Edit” in the Feature Detail Editor Menu. These include *Select All*, *Select*, *Delete*, *Assign to Single or Multiple Viewports*, *Group*, *Ungroup*, *Copy*, *Cut*, *Extract*, and *Merge*.

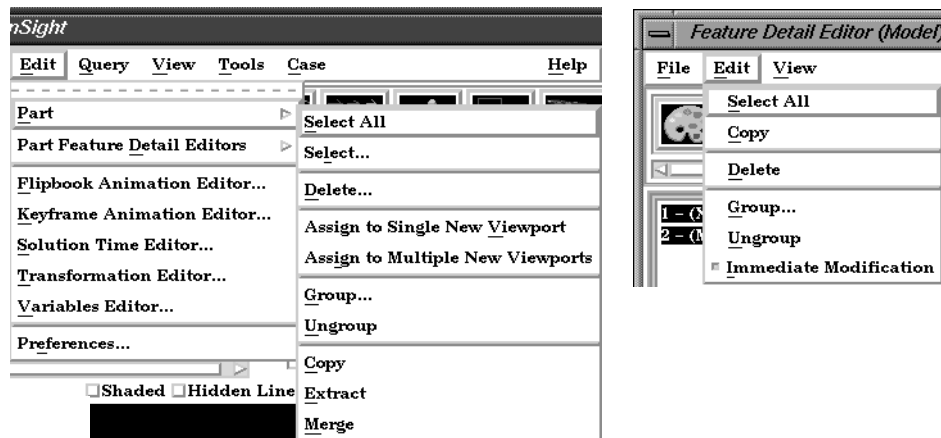


Figure 3-11
Part Operation Selection Menus

Select All

Choosing this from the Main Menu > Edit>Part pull-down, selects all Parts in the Main Parts List. Choosing this from the Edit pull-down in the Feature Detail Editor Menu selects all Parts in the Feature Detail Editor Parts List.

Access: Main Menu > Edit > Part > Select All
Feature Detail Editor Menu > Edit > Select All

(see [How to Select Parts](#))

Select ...

Choosing this from the Main Menu > Edit>Part pull-down, opens the Select Part(s) By Keyword dialog.

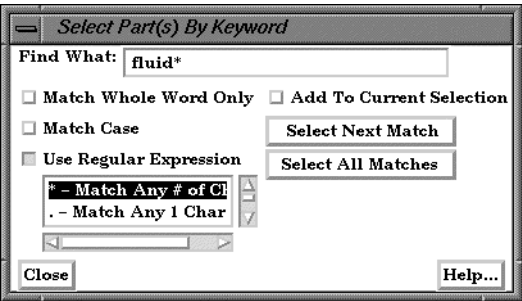


Figure 3-12
Select Part(s) By Keyword dialog

Find What This field is used to specify the keyword or regular expression to compare (match) against Part names.

Match Whole Word Only Toggle When on, the entire Part name must match the keyword or regular expression. When off, a Part name will be selected if only a substring of the Part name matches.

Match Case Toggle When on, the comparison is case sensitive. When off, case is ignored.

Use Regular Expression Toggle	When on, special characters in the keyword will be used to define a regular expression. When off, any special characters will be treated as a regular character during comparison.
Special Character Selection List	<p>Contains a list of special characters available to create a regular expression. Selecting an item from the list will insert the special character into the “Find What” text field at the cursor location.</p> <ul style="list-style-type: none"> * Match any number of characters in the part name . Match any one character in the part name ~ Match part names that do not match the specified search criteria. Separates multiple search keywords or regular expressions. (extra spaces are not allowed around “ ”) <p>Examples:</p> <div style="display: flex; justify-content: space-between;"> <div> <p>Find What: abc*xyz</p> <p><i>Select any Part who's name starts with “abc” and ends with “xyz”</i></p> </div> <div> <p>Match Whole Word Only: On</p> </div> </div> <div style="display: flex; justify-content: space-between;"> <div> <p>Find What: tom jerry</p> <p><i>Select all Part s who's names contain the string “tom” and/or the string “jerry”</i></p> </div> <div> <p>Match Whole Word Only: OFF</p> </div> </div>
Add to Current Selection Toggle	When on, any matching Part names will be added to the list of Part names currently selected. When off, only the matching Part names will be selected.
Select Next Match	Selects the next Part name which matches the keyword or regular expression.
Select All Matches	Selects all Part names which match the keyword or regular expression.
Delete...	<p>If chosen from the Main Menu > Edit > Part pull-down, deletes all selected Parts in the Main Parts List after you have confirmed in a pop-up dialog that you wish to do so. If chosen from the Edit pull-down in the Feature Detail Editor Menu, deletes all selected Parts in the Feature Detail Editor Parts List after you have confirmed in a pop-up dialog that you wish to do so. If model Parts are deleted, they are no longer available for the current session. Parts dependent upon selected Parts will also be deleted or modified</p> <p>Access: Main Menu > Edit > Part > Delete...</p> <p style="padding-left: 40px;">Feature Detail Editor Menu > Edit > Delete...</p> <p>(see How to Delete a Part)</p>
Assign to Single New Viewport	Creates a new viewport and assigns all of the selected parts to the new viewport. The new viewport will be 2D if all of the selected parts are 2D and lie on the same plane.
Assign to Multiple New Viewports	Creates a new viewport for each of the selected parts. Each new viewport will show one part only. If the part is 2D, the viewport will be 2D. Further, if the part assigned to the new viewport is a XYZ or IJK clip or an isosurface, annotation will be created in the lower left corner of the viewport indicating the value of the clip or iso.
Group/Ungroup	<p>The group operation is used to collect any number of parts into a set which can be modified and utilized as one entity. The operation is non-destructive and reversible, and is used solely as a convenience to the user in order to organize a large number of parts.</p> <p>Any attribute modification to a grouped part affects each of the parts in the group. Similarly, if a grouped part is used as a parent part, each part in the group is used as a parent in the creation process.</p>

When group is selected, the dialog shown in the figure below will appear. A part name must be input in order to complete the grouping operation.

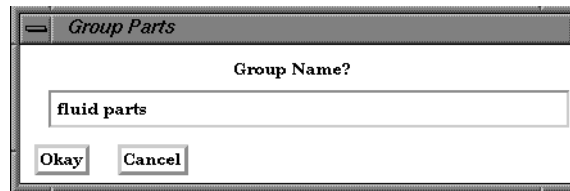


Figure 3-13
Group Parts Dialog

Only parts of the same type and case can be grouped together. Further, groups can not contain other part groups.

Copy (see [How To Group Parts](#))
If chosen from the Main Menu > Edit > Part pull-down, makes a copy of selected Part(s) in the Main Parts List. If chosen from the Edit pull-down in the Feature Detail Editor Menu, makes a copy of selected Part(s) in the Feature Detail Editor Parts List.

The Copy operation creates a dependent copy of another (original) Part. The Copy is created on the Client and its existence is not known to the EnSight Server process. A Copy shares geometric data and variable data with the original Part. (This type of Part is sometimes called a “shallow copy”).

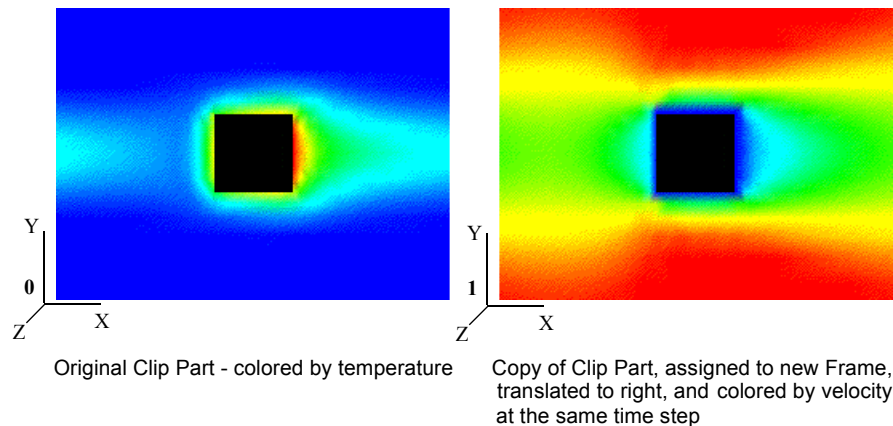


Figure 3-14
Part Copy Example

relationships

The relationship between a Model Part and a Copy made from a Model Part will be one of original and copy. That is, the Model Part will not be a Parent to the Copy as it is to a Created Part such as a clip.

The relationship between a Created Part and a copy made from it will also be one of original and copy since the Copy will initially regard as its Parent the same Part that the original Created Part regards as its Parent. The Parent of individual Created Parts can of course be reassigned (using the Update Parent button at the bottom of the Feature Detail Editor) but the Parent of a Created Part Copy can Not be reassigned.

A copy can be used as a Parent Part for Parts created since the create operation will operate on the original Part.

attributes	The initial attributes assigned to a Copy are the same as those of the original Part at the time of copying. All attributes for the Copy except Element Representation (3D border, 2D full, border, Feature Angle, etc.) can be changed. The Element Representation of a Copy cannot be changed independently; a change in Element Representation of the original changes the copy as well.
description	The description of the new Copy will be the same as the original Part with the suffix “-COPY” added (of course, you can change this description in the Desc field in the Feature Detail Editor).
copies of copies	You can make multiple copies from a Model or Created Part, but you can Not make copies of copies.
frame assignment	A new frame is automatically created for each newly created Copy and the Copy is assigned to the new frame so that it can easily be moved with a local transformation. The location of the original Part and the Copy will initially coincide as well. Like all Parts, Copies of Parts can be reassigned to different frames in the General Attributes Section of the Feature Detail Editor (for that type of Part).
usefulness	One of the most useful purposes for copies is a separation allowing for the side-by-side display of different attributes (shown in Figure 3-11). Since all attributes except Element Representation can be different, the original and the copy can be displaying different variables, different displacements, etc.

(see [How To Copy a Part](#))

Extract

Extracts selected Part(s) into a new, true Part, *using the Part representation in effect at the time* (full, border, or feature). If more than one Part is selected, then they are joined into a single Part. If more than one Part is selected when extract is invoked, then all will have their extracted geometry joined into a single new Part. The new Part is assigned to Frame 0.

The Extract option is closely tied to Element Representation. It creates a new Part using the geometry of the current representation (what you see is what you get). Extracted Parts which are in Full Representation are actual copies of the original, but extracted Parts which are in Border Representation are only the shell or boundary of the original. Extract is often used with the Save Geometric Entities feature to save extracted Parts (and not the originals) into a smaller set of data. It is also used to create hollow Parts from solid Parts to be able to look inside a solid Part after cutting it open with the Cut feature.

(see [How To Extract Part Representations](#))

Merge

If more than one Part is selected, the Merge operation creates a new model Part on the Server host that is a combination of all selected. If only one Part is selected when Merge is invoked, then a new Part is created on the Server host that is identical but fully independent from the original Part (Note that this type of “copy” does not have the restriction on Element Representation that Part Copy does, - *all* Attributes can be reassigned - but it requires considerably more memory because it does not share the geometry with the original but now has its own copy of the geometry). The merge operation creates a new Part. The new Part is assigned the default Display Attributes and is also assigned to Frame 0.

(see [How To Merge Parts](#))

